A Model for Subgrid Phase Interface Dynamics due to Surface Tension

M. Herrmann*
School for Engineering of Matter, Transport and Energy
Arizona State University, Tempe, AZ 85287

Abstract
This paper reports on the development of a model for the sub-filter surface tension-induced motion of liquid/gas phase interfaces. A key feature of the proposed model is to take the sub-filter phase interface dynamics fully into account by employing a dual-scale approach. Instead of modeling the sub-filter phase interface geometry, it is resolved on an auxiliary grid using the Refined Level Set Grid approach. The required sub-filter velocity field on the auxiliary grid is reconstructed solving a PDE in a narrow band surrounding the phase interface. With the fully resolved phase interface geometry available, all previously unclosed terms in the filtered Navier-Stokes equations can be directly closed using explicit filtering. The model is successfully applied to the simulation of subgrid oscillating drops and the subgrid capillary breakup of liquid ligaments.

*Corresponding Author: marcus.herrmann@asu.edu
Introduction

Atomization of liquids is a key component in many natural phenomena and technical processes. Combustion in most engines, for example, relies on the liquid fuel being atomized, evaporated, and mixed with air before energy conversion in the form of chemical reactions can occur. Since engine performance, efficiency, and pollutant production strongly depend on the quality of the gaseous fuel/air mixture, the details of the atomization process are of paramount importance in achieving sustainable energy conversion. Unfortunately, liquid fuel atomization remains poorly understood. This is in part due to the fact that experimental access to the initial breakup processes, the so-called primary atomization, is extremely challenging. Some progress has been made in recent years using X-ray phase contrast imaging [1, 2] and ballistic imaging [3]; however, the data available from these techniques are not yet sufficient to study the phase interface dynamics in detail. As an alternative approach, detailed numerical simulations of the phase interface dynamics during primary atomization, resolving all relevant time and length scales, can be used to study the phase interface dynamics [4, 5, 6].

However, due to the vast range of length and time scales involved in the primary atomization process, the computational resources required to perform detailed simulations are enormous, taxing even the most powerful computers available today [7]. Since primary atomization is governed by the dynamics of the phase interface, a need therefore exists for appropriate phase interface dynamics models that make the computational cost of predicting the primary atomization outcome more tractable.

A wide range of phenomenological models aiming to represent statistically the essential features of primary atomization have been proposed. Although conventional phenomenological models introduce the mechanisms of the primary atomization of the liquid jet, i.e., surface instabilities [8, 9, 10, 11], drop shedding [12], spontaneous breakup [13], and jet turbulence [14], they use round blobs injected from the nozzle exit and hence neglect all details of the phase interface dynamics.

Other modeling approaches to primary atomization include the stochastic model by [15, 16] representing the phase interface by constituent stochastic particles and the mean interface density transport equation model for Reynolds-averaged Navier-Stokes (RANS) approaches by [17] and [18]. The former treats the phase interface dynamics in a stochastic sense but requires the a priori knowledge of the breakup mechanism, the latter is affected by the drawbacks of the RANS approach: the transport of the mean interface density is modeled by a diffusion-like hypothesis, thereby neglecting the spatial groupings of liquid elements [7]. Furthermore, it is well known that flows with the complexity occurring in actual atomization cases, such as aircraft engine combustors or augmentors, which can exhibit swirling flows, multiple recirculation regions, and jets in cross-flow, are not predicted accurately by RANS methods.

Large-eddy simulations (LES), on the other hand, have been demonstrated to be very accurate for swirling flows [19, 20] and flows with strong recirculation regions [21, 22, 23]. A model describing the primary atomization process should thus be consistent with the LES approach, in order to be applicable to real-world engineering applications.

The purpose of this paper is to present a sub-filter model for the phase interface dynamics of atomizing flows in the context of LES. It follows the dual-scale modeling approach presented in [24, 25]; however, the focus will be on the contribution of the sub-filter surface tension force only.

The paper is structured in the following manner. After presentation of the governing equations and derivation of the sub-filter model, the numerical implementation of the model is discussed. Following, simulation results are presented and discussed before conclusions are summarized.

Governing equations

The equations governing the fully resolved motion of an unsteady, incompressible, immiscible, two-fluid system are the Navier-Stokes equations,

\[
\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \frac{1}{\rho} \nabla \cdot (\mu (\nabla \mathbf{u} + \nabla^T \mathbf{u})) + \frac{1}{\rho} \mathbf{T}_\sigma,
\]

where \( \mathbf{u} \) is the velocity, \( \rho \) the density, \( p \) the pressure, \( \mu \) the dynamic viscosity, and \( \mathbf{T}_\sigma \) the surface tension force which is non-zero only at the location of the phase interface \( \mathbf{x}_f \). Furthermore, the continuity equation results in a divergence-free constraint on the velocity field,

\[
\nabla \cdot \mathbf{u} = 0.
\]

The phase interface location \( \mathbf{x}_f \) between the two fluids can be described by a level set scalar \( G \), with

\[
G(\mathbf{x}_f, t) = 0
\]

at the interface, \( G(\mathbf{x}, t) > 0 \) in fluid 1, and \( G(\mathbf{x}, t) < 0 \) in fluid 2. Differentiating Eq. (3) with respect to time yields the level set equation,

\[
\frac{\partial G}{\partial t} + \mathbf{u} \cdot \nabla G = 0.
\]
Assuming \( \rho \) and \( \mu \) are constant within each fluid, density and viscosity at any point \( x \) can be calculated from

\[
\begin{align*}
\rho(x) &= H(G)\rho_l + (1 - H(G))\rho_g \quad (5) \\
\mu(x) &= H(G)\mu_l + (1 - H(G))\mu_g, \quad (6)
\end{align*}
\]

where indices \( l \) and \( g \) denote values in liquid and gas, respectively, and \( H \) is the Heaviside function. From Eq. (3) it follows that

\[
\delta(x - x_f) = \delta(G)|\nabla G|, \quad (7)
\]

with \( \delta \) the Dirac delta function. Furthermore, the interface normal vector \( n \) and the interface curvature \( \kappa \) can be expressed in terms of the level set scalar as

\[
n = \frac{\nabla G}{|\nabla G|}, \quad \kappa = \nabla \cdot n. \quad (8)
\]

Using Eqs. (7) and (8), the surface tension force \( T_\sigma \) can thus be expressed as

\[
T_\sigma(x) = \sigma \kappa \delta(x - x_f)n = \sigma \kappa \delta(G)|\nabla G|n, \quad (9)
\]

with \( \sigma \) the constant surface tension coefficient.

**Filtered governing equations**

Introducing spatial filtering into Eqs. (1) and (2), the filtered governing equations can be derived,

\[
\frac{\partial \tilde{u}}{\partial t} + \tilde{u} \cdot \nabla \tilde{u} = -\frac{1}{\tilde{\rho}} \nabla \tilde{p} + \frac{1}{\tilde{\rho}} \nabla \cdot (\tilde{\tau} + \tilde{\tau}^SGS) \\
+ \frac{1}{\tilde{\rho}} T_\sigma, \quad (10)
\]

\[
\nabla \cdot \tilde{u} = 0, \quad (11)
\]

where \( \tilde{\cdot} \) denotes Favre filtering, \( \tilde{\rho} = \rho \tilde{f} / \rho, \) \( \tilde{\rho} \) indicates spatial filtering, \( \tilde{\tau} \) is the stress tensor calculated using filtered velocities, and \( \tilde{\tau}^SGS \) is the sub-filter contribution of the convective term. Using Eqs. (5) and (6), the filtered density and viscosity are

\[
\begin{align*}
\tilde{\rho} &= \rho_l \int G(x)H(G)dx \\
&+ \rho_g \int G(x)(1 - H(G))dx \quad (12) \\
\tilde{\mu} &= \mu_l \int G(x)H(G)dx \\
&+ \mu_g \int G(x)(1 - H(G))dx, \quad (13)
\end{align*}
\]

where \( G \) is a normalized spatial filter function. Defining

\[
\tilde{\psi} = \int G(x)H(G)dx, \quad (14)
\]

these equations reduce to

\[
\begin{align*}
\bar{\rho} &= \rho_l \bar{\psi} + \rho_g(1 - \bar{\psi}) \quad (15) \\
\bar{\mu} &= \mu_l \bar{\psi} + \mu_g(1 - \bar{\psi}), \quad (16)
\end{align*}
\]

which requires a model for \( \bar{\psi} \). The filtered surface tension term in Eq. (10) is

\[
T_\sigma = \sigma \kappa \delta(x - x_f)n. \quad (17)
\]

In summary, three terms in the above governing equations are unclosed and require modeling: \( \bar{\psi}, \tau^SGS, \) and \( T_\sigma \).

**The dual-scale approach to modeling sub-filter phase interface dynamics**

Instead of relying on a cascade process by which dynamics on a sub-filter scale can be inferred from the dynamics on the resolved scale, the dual-scale approach proposed in [24, 25] aims to maintain a fully resolved realization of the phase interface geometry at all times, expressed, for example, in terms of a level set scalar \( G \). Then \( \bar{\psi} \) can be calculated exactly by explicit filtering of \( H(G) \) using Eq. (14). This in turn would give \( \bar{\sigma} \) and \( \bar{\mu} \) exactly using Eqs. (15) and (16).

Using a standard single-phase model for the the sub-filter contribution of the stress tensor \( \tau^SGS \), the only term left to close is the filtered surface tension term \( T_\sigma \). However, using Eq. (17) and realizing that \( \kappa, x_f, \) and \( n \) are simply functions of \( G \) and thus available as a fully resolved realization, \( T_\sigma \) can be calculated exactly by simple application of the spatial filter function,

\[
T_\sigma = \int G(x)\sigma \kappa \delta(x - x_f)n dx. \quad (18)
\]

Although the above are exact closures, the problem of modeling is of course simply shifted to the problem of maintaining a fully resolved realization of the phase interface geometry, i.e., describing the fully resolved motion of the phase interface. Since \( u = \tilde{u} + u_{sg} \), this results in

\[
\frac{\partial G}{\partial t} + (\tilde{u} + u_{sg}) \cdot \nabla G = 0, \quad (19)
\]

where the only term requiring modeling is \( u_{sg} \).

In [24, 25] a model for \( u_{sg} \) is proposed consisting of three contributions,

\[
u_{sg} = u' + \delta u + u_\sigma, \quad (20)
\]

where \( u' \) is due to sub-filter turbulent eddies, \( \delta u \) is attributed to the interface velocity increment due to relative sub-filter motion between the two phases,
and $\mathbf{u}_\sigma$ is due to sub-filter velocities induced by sub-filter surface tension forces. The focus of the current paper is on the last term; for modeling outlines of the first two, the reader is referred to [24, 25].

A model for sub-filter velocities induced by sub-filter surface tension

We limit ourselves in the following to flows where the turbulence and the viscous boundary layers at the phase interface are resolved on the flow solver, i.e., the first two terms of Eq. (20) can be neglected, i.e., $\mathbf{u}_{sg} = \mathbf{u}_\sigma$. Differentiating this with respect to time results in $D\mathbf{u}_{sg}/Dt = \mathbf{a}_\sigma$, where $\mathbf{a}_\sigma$ is the sub-filter acceleration of the phase interface due to sub-filter surface tension forces. Since $\mathbf{u}_{sg}$ is a property of the phase interface at $G = 0$, using Eq. (19) the transport equation for $\mathbf{u}_{sg}$ follows as

$$\frac{\partial \mathbf{u}_{sg}}{\partial t} + (\mathbf{u} + \mathbf{u}_{sg}) \cdot \nabla \mathbf{u}_{sg} = \mathbf{a}_\sigma. \tag{21}$$

Note that Eq. (21), like Eq. (19), is valid only at the location of the fully resolved phase interface $G = 0$, and because of Eq. (2) and $\nabla \cdot \mathbf{u} = 0$,

$$\nabla \cdot \mathbf{u}_{sg} = 0. \tag{22}$$

The model for $\mathbf{a}_\sigma$ follows the Taylor analogy of a spring/damper system [24, 25], resulting in

$$\mathbf{a}_\sigma = \mathbf{a}_\kappa + \mathbf{a}_\mu = \frac{1}{\text{We}}(\kappa - \overline{\kappa})n\delta - \frac{1}{\text{Re}}\mathbf{u}_\sigma, \tag{23}$$

where $\kappa$ is the fully resolved curvature, $\overline{\kappa}$ is the explicitly filtered curvature,

$$\overline{\kappa} = \int \mathcal{G}(x)\kappa\delta(x - x_f)dx, \tag{24}$$

and We and Re are a model Weber and Reynolds number, respectively. The first term in Eq. (23), $\mathbf{a}_\kappa$, acts in essence as a spring, whereas the second term, $\mathbf{a}_\mu$, acts as a damper since on the small scales considered here, viscous forces are important.

Numerical methods

To efficiently solve Eqs. (19) and (21) for the fully resolved phase interface, the Refined Level Set Grid (RLSG) method [26] is used. By design, it solves the level set equation on a separate, highly resolved Cartesian G-grid, independent of the underlying flow solver grid, using a dual narrow-band approach. The numerical approach is optimized for massively parallel computer systems and allows for significantly refined G-grids at low cost compared to the flow solver cost [26].

Coupling to a finite volume flow solver is achieved by explicitly evaluating the definition of a control volume quantity $\phi_{cv}$,

$$\phi_{cv} = \frac{1}{V_{cv}} \int_{V_{cv}} \phi(x)dx, \tag{25}$$

using $\phi$ defined on the fully resolved G-grid. Note that this is in fact equivalent to applying a spatial filter to the fully resolved equation, where the filter size is the local flow solver control volume $V_{cv}$. Standard LES approaches use this grid filter as their implicit filter, solving for the filtered quantities directly on a given grid. We thus propose to use Eq. (25) as the spatial filter function $\mathcal{G}$, resulting in

$$\overline{\psi}_{cv} = \frac{1}{V_{cv}} \int_{V_{cv}} H(G(x))dx, \tag{26}$$

$$\overline{\kappa}_{cv} = \frac{1}{A_{cv}} \int_{V_{cv}} \kappa\delta(x - x_f)dx, \tag{27}$$

where

$$A_{cv} = \int_{V_{cv}} \delta(x - x_f)dx. \tag{28}$$

In principle, the filtered surface tension force can be evaluated as

$$\overline{T}_{\sigma,cv} = \frac{1}{A_{cv}} \int_{V_{cv}} \sigma\kappa\delta(x - x_f)n \cdot dx; \tag{29}$$

however, to be consistent with the balanced force approach [26], we choose to model $\overline{T}_{\sigma}$ as

$$\overline{T}_{\sigma,cv} = \sigma\overline{\kappa} \nabla \overline{\psi}_{cv}. \tag{30}$$

Note that the above filter operations are consistent with the scaling symmetries of the level set equation [27].

To evaluate Eq. (26), we use the methodology described in [26] based on analytical formulas derived by [28]. Since Eq. (27) contains a delta function affixed to the phase interface, it constitutes a surface filter of curvature. It is evaluated by first triangulating the $G = 0$ phase interface using a marching cubes algorithm on the G-grid and then replacing the integral by the sum over the surface triangles using G-grid nodal curvatures interpolated to the triangle centers.

Equation (21) is defined only at the location of the phase interface $G = 0$. We thus define each spatial component of $\mathbf{u}_{sg}$ to be constant in the phase interface normal direction, by solving

$$\nabla \mathbf{u}_{sg,i} \cdot \nabla G = 0, \quad i = 1..3 \tag{31}$$

using the PDE-based iterative redistribution scheme of [29] with first-order spatial and temporal discretization.
Enforcing Eq. (22) ensures that the liquid volume enclosed by the $G = 0$ iso-surface is conserved during interface motion by the sub-filter velocity $u_{sg}$. Instead of enforcing it locally on every $G$-grid cell, which would entail a projection/correction algorithm solving a Poisson system, we enforce the liquid conservation condition either globally, within each filter volume, or locally on the basis of individual separated liquid structures. In each case, we first calculate the change of liquid volume due to $u_{sg}$ by integrating

$$\alpha_{sg} = \frac{1}{A} \int_V u_{sg} \cdot n \, \delta(x - x_f) \, dx,$$

where $V$ is either the global volume, the spatial filter volume, or the volume enclosing a single separated liquid structure, and $A$ is the phase interface surface area contained therein. The liquid volume in $V$ is conserved, if $\alpha_{sg} = 0$, and thus we correct $u_{sg}$ by subtracting $\alpha_{sg} n$ at the end of every time step. Equation (32) is evaluated using the marching cubes surface triangle summation algorithm. If needed, individual separated liquid structures are identified using the parallel drop identification algorithm proposed in [30].

The algorithm for the sub-filter model is thus as follows:

1. filter curvature using marching cubes surface integrals, Eq. (27)
2. for each $G$-node $i \in V_{cv}$ set $\kappa_i = \kappa_{cv}$,  
3. solve sub-filter velocity PDE (21),  
4. redistribute each spatial component of $u_{sg}$ in the front normal direction, Eq. (31),  
5. enforce sub-filter motion liquid volume conservation correcting $u_{sg}$ using Eq. (32).

Note that due to the dual-narrow band structure of the RLSG method, all model equations need to be solved only in the close vicinity of the phase interface.

**Results**

In this section, results obtained using the model for sub-filter surface tension-induced phase interface dynamics are presented. In the first three tests, the entire phase interface is embedded within a single flow solver cell, i.e., all phase interface dynamics occur within a single spatial filter. The last test shows the applicability of the proposed model to the case when part of the dynamics are resolved by the flow solver, and parts need to be modeled by the subgrid model.

*Sub-filter oscillating drop*

In this test, a mode two deformed drop of radius $R_0 = 0.2$ is placed in the center of a single unit-sized flow solver cell. The distance $R$ of the drop’s surface to its center as a function of the latitude $\phi$ is given as

$$R(\phi) = R_0 + \frac{1}{2} A_0 \left( 3 \cos^2 \phi - 1 \right),$$

with $A_0 = 0.2$. The three-dimensional sub-filter volume, i.e., the flow solver cell, is discretized by $G$-grid cells of size $\Delta x_G = 1/64$. Equation (22) is enforced solving Eq. (32) within the flow solver spatial filter volume and the model Weber and Reynolds numbers are both set to unity.

Figure 1 depicts the distance $R$ from the drop’s center to its surface along the x-axis (black) and y-axis (gray).

*Sub-filter Rayleigh-Plateau stable column*

In this test, a liquid column inside a periodic unit-sized box is subjected to a strong sinusoidal perturbation of amplitude $A_0 = 0.05$ and wavelength $\lambda_0 = 0.2$ that is stable according to the Rayleigh-Plateau instability mechanism. In fact, the
liquid column radius \( R_0 = 0.2 \) is chosen such that \( 2\pi R_0/1 = 1.26 > 1 \), i.e., all possible perturbation wavelengths are stable and should decay. Again, only a single flow solver cell is used for the entire computational domain, discretized by a \( G \)-grid of size \( \Delta x_G = 1/64 \). Equation (22) is enforced solving Eq. (32) within the flow solver spatial filter volume and the model Weber and Reynolds numbers are both set to unity.

Figure 2 shows the temporal evolution of the disturbance amplitude. Note that the high-frequency component that corresponds to the short wavelength initial perturbation is quickly damped, leaving only the slowly decaying disturbance with wavelength equal to the box length. As expected, the column settles down to a stable configuration for long times. Figure 4 shows snapshots of the phase interface geometry colored by the sub-filter velocity component in the \( y \)-direction (vertical direction). Note that if no subgrid model is used, the initial sinusoidal perturbation of the column would incorrectly persist for all times and no breakup would occur. Using the subgrid model, however, correctly breaks the column into two, equally sized drops that undergo brief damped oscillations.

**Capillary breakup of a ligament**

This case tests the coupling between flow resolved by the flow solver mesh and the subgrid model. A liquid column of initial radius \( R_0 = 0.1 \) is perturbed by a cosine wave of wavelength \( \lambda_0 = 1 \) and initial amplitude of \( A_0 = 0.02 \) and placed inside a periodic box of size \( L \) by \( L/2 \) by \( L/2 \) with \( L = 1 \). Material properties between the two phases are matched with \( \rho = 1, \mu = 0.01 \), and \( \sigma = 0.1 \).

Figure 6 shows in the first column the results of a high-resolution reference solution obtained using a flow solver resolution of \( \Delta x = 1/128 \) and \( G \)-grid resolution of \( \Delta x_G = 1/256 \) without the subgrid model. Shortly after \( t = 1 \) the thin liquid ligaments formed on either side of the central large drop break close to the drop’s surface. The remaining ligaments quickly (within about \( \Delta t = 0.18 \)) pull back forming a second small satellite drop. Both drops undergo oscillations, with the smaller one oscillating with higher frequency and coming to rest much sooner than the larger drop, which continues to oscillate slightly up to the end of the simulation at \( t = 2.5 \).

The second column of Figure 6 depicts results using a flow solver resolution of \( \Delta x = 1/16 \) and the same \( G \)-grid resolution of \( \Delta x_G = \Delta x \). The column breaks up significantly earlier at around \( t = 0.1 \) without forming any stretched-out ligaments that could result in secondary drops. Only a large single drop remains that undergoes a single period of oscillations before coming to rest.

The third column shows the results of refining the \( G \)-grid to \( \Delta x_G = 1/128 \), i.e., 512 \( G \)-grid nodes per flow solver control volume, but without activating the proposed subgrid model. Qualitatively some features of the reference solution are retained, that is a thin ligament is formed connected to a single large drop. The ligament pinches off near the main drop at around \( t = 1.7 \), hence significantly later than in the reference solution. The ligament then very slowly pulls back. This pullback is mostly due to numerical errors causing local liquid volume loss. Even after \( \Delta t = 0.8 \) the ligament has not yet pulled fully.
back into the secondary drop. Comparing this result to the coarse grid solution (second column), it is evident that the lack of a connecting ligament in the coarse grid results is due to numerical errors in tracking the phase interface. Refining the G-grid, as done in column three, reduces interface tracking errors such that the ligament is retained. However, reducing the phase interface tracking errors without the proposed subgrid model induces severe surface deformations evident on the main drop. Surface tension should smooth these; however, in this case surface tension is active only on the flow solver scale and thus perturbations smaller than $\Delta x$ can persist unimpeded.

Finally, the last column of Figure 6 depicts the results obtained using the proposed subgrid model with $u_{ref} = 1$ and thus $We = \rho u_{ref}^2 L/\sigma = 10$ and $Re = \rho u_{ref} L/\mu = 100$, employing the same grid resolution as in the previous column. Again a thin connecting ligament is formed which correctly pinches at the main drop, however at a slightly delayed time $t = 1.36$. The ligament then pulls back at a slower rate than in the reference solution such that two additional satellite drops are formed on either end of the ligament. Due to numerical errors, these disappear leaving only the main and satellite drop; however, they could be maintained by transferring them into a Lagrangian representation as proposed in [30]. No unphysical perturbations on the drops can be observed. As in the reference solution, the two generated drops undergo damped oscillations, with the small drop coming to rest before the large drop, which continues to oscillate until the end of the simulation.

Overall, the results using the subgrid model are significantly better than those obtained using either no finer G-grid (second column) or using a finer G-grid without a subgrid model (third column). The generation of additional satellite drops compared to the reference solution is likely due to non-negligible dissipative errors in solving Eq. (31) with a first-order method. This acts as an additional viscous term in Eq. (23) impeding the fast pullback of the very thin ligament. Because the ligament pulls back slower than in the reference case, there is sufficient time for a capillary instability to generate the additional satellite drops. Switching to a higher-order iterative method to solve Eq. (31) or to a fast marching method should remedy this problem. Comparing computational cost, the solution obtained using the subgrid model is more than a factor 100 faster than the reference solution, indicating that the proposed method has the potential to lower computational cost when compared to a full direct numerical simulation.

Conclusions

In this paper, a novel subgrid model for the sub-filter surface tension induced motion of phase interfaces is presented. The model is based on a dual scale approach that allows for exact closure of the unclosed terms in the Navier-Stokes equations by explicitly filtering the fully resolved phase interface. The model shows promising results in tests of oscillating drops, Rayleigh-Plateau instability, and the viscous breakup of a liquid column. Further verification and validation is, however, required and will be pursued in the future, together with expanding the proposed model to take turbulence into account.

Acknowledgments

Support from the Center for Turbulence Research’s Summer Program and the National Science Foundation under grant CBET-0853627 is gratefully acknowledged. The author would also like to thank O. Desjardins and M. Gorokhovski for many fruitful discussions.

References


Figure 3. From left to right: surface tension acceleration $a_\kappa \cdot n$ of Eq. (23), viscous acceleration $a_\mu \cdot n$ of Eq. (23), subgrid x-velocity, and subgrid z-velocity at $t = 0.05, 0.1, 0.2, 0.3, \text{ and } 0.6$ (top to bottom) for oscillating drop case.
Figure 4. Subgrid z-velocity for Rayleigh-Plateau stable column at $t = 0.0, 0.05, 0.1, 0.15, 0.2, 0.5, 1, 2, \text{ and } 7$ (top left to bottom right)

Figure 5. Subgrid z-velocity for Rayleigh-Plateau stable column at $t = 0.0, 0.1, 1, 1.2, 1.26, 1.3, 1.4, 1.5, \text{ and } 7$ (top left to bottom right)
Figure 6. Breakup of a liquid viscous column: reference solution, flow solver resolution only, flow solver and G-grid sub-filter resolution without subgrid model, and flow solver and G-grid sub-filter resolution with subgrid model (from left to right) at $t = 0, 0.14, 0.9, 1.0, 1.14, 1.36, 1.5, 1.7, 2.0, \text{ and } 2.5$. 